

Heat Transfer around a Square Cylinder in Unsteady Flow Regime

In partial fulfilment of the requirements of

Bachelor of Technology

in

Chemical Engineering

Submitted By

Eugene Tete (Roll No.110ch0091)

Under the guidance of

Dr. Akhilesh Kumar Sahu



DEPARTMENT OF CHEMICAL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY

ROURKELA

ODISHA, 2015

CERTIFICATE

This is to certify that the thesis entitled “Heat Transfer around a Square Cylinder in Unsteady Flow Regime” submitted by Eugene Tete (110ch0091) in partial fulfilment of the requirements for the award of degree of Bachelor of Technology in Chemical Engineering at National Institute of Technology, Rourkela is an authentic work carried out by him under my supervision and guidance.

Date

Place: Rourkela

Dr. Akhilesh Kumar Sahu

Department of Chemical Engineering,

National Institute of Technology,

Rourkela- 769008

ACKNOWLEDGEMENT

With a feeling of great pleasure, I express my sincere gratitude to Dr. Akhilesh Kumar Sahu for his superb guidance, support and constructive criticism, which led to the improvements and completion of this project work.

Many special thanks go to all of the people that involve in this study for their excellent co-operation, inspiration and supports during the study. My sincere goes to our Hod sir, teachers and my companions who helped me in many ways. My lab mates Amogh, Pritish and Bishmay helped me a lot in completing this project. I would like to thank Trupti Ranjan Behera for all the software help.

Eugene Tete(Roll no: 110ch0091)

Department of Chemical Engineering

National Institute of Technology, Rourkela

ABSTRACT

The project report deals with the analysis of fluid flow phenomena at different conditions utilizing the CFD simulation software ANSYS. This project is to study the simulation and analysis problems that involve fluid flows. First of all we get acquainted with the CFD software. In this project we are using ANSYS-FLUENT for analysing the mathematical model of a physical problem. Simplifying assumptions are made in order to make the problem tractable (e.g., steady state, incompressible, in viscid, two dimensional). The project consists of two parts. In the first part we observe the lid driven square cavity using water as the material. The work was done for both Newtonian ($n=1$) and shear-thinning Non-Newtonian fluid ($n=0.5$, $k=5$). The simulations were carried out at different Reynolds Number ($100 < Re < 1000$).

The second part is the study of two dimensional unsteady laminar flow of a Newtonian fluid ($n=1$) across a square cylinder. The study was done with the varying Reynolds Number (90, 100 and 120) at Prandlt Number taken to be (0.7, 5, 10, and 20). Having established the limits of the unsteady flow regime Drag and Nusselt Number results are obtained. The values of Drag coefficient, Lift coefficient, Nusselt Number was plotted versus the flow time. The Nusselt Number shows positive dependence on both the Reynolds Number and Prandlt Number. The streamline pattern and isothermal pattern was also shown.

CONTENTS

Chapters		Topic	Page no.
		Cover page	i
		Certificate	ii
		Acknowledgement	iii
		Abstract	iv
		Contents	v
		List of Figures	vi
		List of tables	vii
		Nomenclature	viii
Chapter 1		Introduction	1
	1.1	Newtonian fluid	1
	1.2	Non-Newtonian fluid	2
	1.3	Classification of fluids	2
	1.4	Computational Fluid Dynamics	2
	1.5	Drag coefficient	2
Chapter 2		Literature Review	3
	2.1	CFD	3
	2.2	Advantages of CFD	4
	2.3	Basic review on heat transfer	4
Chapter 3		Mathematical Formulation	6
Chapter 4		Results and Discussion	10
	4.1	Problem Statement 1	10
	4.2	Problem Statement 2	19
Chapter 5		Conclusion	28
		References	29

LIST OF FIGURES

Sl no.	Title	Fig no.	Page no.
1	Schematic flow around a square cylinder	1	6
2	Lid driven cavity	2	10
3	Stream function for $100 < Re < 1000$ in a square cavity for a Newtonian fluid	3	11
4	Horizontal centre line velocity for $100 < Re < 10,000$	4	13
6	Vertical centre line velocity for $100 < Re < 10,000$	5	15
7	Comparison of Horizontal centre line velocities and Vertical centre line velocities for $Re=100$ & 400 by Ghia et al for Newtonian fluid	6-7	17
8	Comparison of Horizontal centreline velocities and Vertical centre line velocities for $Re=100$ by Neotyfou for Non-Newtonian fluid	8-9	18
9	Flow over a square cylinder	10	19
10	Isothermal pattern at $Pr=0.7$ for various Reynolds number	11	23
11	Stream function at $Pr=0.7$ for various Reynolds number	12	24
12	Drag coefficient and Lift coefficient variation on square cylinder for different Reynolds number at $Pr=0.7$	13	25
13	Nusselt number variation on square cylinder surface at different Prandlt number	14	26
14	Variation of average Nusselt number with Re and Pr	15	27

LIST OF TABLES

Sl no.	Title	Table no.	Page no.
1	Values of Drag coefficient and Nusselt no at $Pr=0.7$	1	21
2	Values of Drag coefficient and Nusselt no at $Pr=5$	2	21
3	Values of Drag coefficient and Nusselt no at $Pr=10$	3	22
4	Values of Drag coefficient and Nusselt no at $Pr=20$	4	22

NOMENCLATURE

C_L = Lift coefficient

C_D = Drag coefficient

D = Diameter of the square cylinder

F_D = Drag force on cylinder

F_L = Lift force on cylinder

Re = Reynolds number

U = Component of velocity in X-direction

V = Component of velocity in Y-direction

C_p = Specific heat of the fluid

k = Thermal conductivity of the fluid

Nu = Local Nusselt number

μ = Dynamic viscosity of the fluid

ρ = Density of the fluid

h = Local convective heat transfer coefficient

\overline{Nu} = Average Nusselt number of the cylinder

Pr = Prandtl number

θ = Non-dimensional temperature

t = Time

T = Temperature

T_w = Constant cylinder temperature

T_∞ = Uniform temperature of the fluid

P = Pressure

τ = Non-dimensional time

B = Size of the cylinder

H = Height of the computation domain

L = Length of the computational domain

u_∞ = Uniform velocity of the fluid

n = Cylinder surface normal direction

u = Velocity of the fluid

CHAPTER 1

INTRODUCTION

The bluff bodies like circular cylinder ,rectangular prism etc. are the most common engineering structural configurations resembling buildings, bridges, chimneys, cooling towers etc. Engineers quite face various flow-induced problems, in dealing with the flow around these bodies, most common of which include the flow-induced vibrations. The flow induced vibrations arise due to a very complex phenomenon called vortex shedding which has been addressed quite extensively. The bluff bodies actually create a turgid region consisting of the separated flow and extensively forms huge unsteady wake region in the downstream. The unsteady lift and drag forces are usually generated by the vortex shedding so formed in the wake region of these bodies. The vortices undergo alternate shedding whenever the fluid flows around the square cylinder and hence Karman Vortex Street is formed representing the wake behaviour of the cylinder.

Vortex shedding causes the periodic vibrations of the square cylinder and the other structures in transverse flow. Damaging oscillations may also occur due to steady flow across the square cylinder. In such cases the shedding frequency of the vortices is almost close to the natural frequency of the obstacle. The fluid flow patterns of the wake region are formed behind the square cylinder due to alternate deflections. This causes the induced forces on the square cylinder to become more periodic and culminate in conjunction between the fluid and the structure. This is observed through the oscillation behaviour of the square cylinder. If the resulting excitation frequency contemporizes with the corresponding frequency of the square cylinder then the resonance is observed. Hence simulation of unsteady flow past a square cylinder is practically relevant.

Usually the flow over a square cylinder is restricted within a channel or subject to a free stream flow. Within such flows various physical phenomena occurs such as flow separation, reattachment, recirculation and vortex shedding are established, producing a very challenging flow field for both experimentalist and CFD users.

Understanding the wake behaviour and associated dynamics of flow past a square cylinder helps in the better design of the concerned or desired objectives where the engineering parameters need to be designed with reasonable practices.

1.1 Newtonian Fluid: A fluid whose stress at each point is linearly proportional to its strain rate at that point. It obeys Newton's law of viscosity and for which μ has a constant value

$$\tau = \mu \frac{\partial u}{\partial y}$$

τ is shear stress given in the fluid.

μ is the scalar constant of proportionality often considered as the fluid viscosity and,

$\frac{\partial u}{\partial y}$ is the derivative of the velocity component, it is parallel to the direction of shear and is relative to displacement in perpendicular direction.

1.2 Non-Newtonian Fluid: It is a fluid whose viscosity is variable based on applied stress. Non-Newtonian fluids help to understand the wide variety of fluids that exist in the physical world. Plastic solids, power law fluids, viscoelastic fluids and time dependant viscosity fluids are others that exhibit complex relationship between shear stress and viscosity. It does not obey Newtonian's law of viscosity. In case of such a fluid, the plot between shear stress and shear rate is different and can even be time dependent.

$$\tau \neq \mu \frac{\partial u}{\partial y}$$

1.3 Classification of fluids:

Newtonian fluids $\tau = \mu \frac{\partial u}{\partial y}$ e.g. air, water and many other engineering fluids behave as Newtonian fluids at normal circumstances.	Non Newtonian fluids $\tau \neq \mu \frac{\partial u}{\partial y}$		
	Purely viscous fluids		Viscoelastic fluids
	Time-Independent	Time-dependant	
	(i) Pseudoplastic fluids $\tau = \mu \left(\frac{\partial u}{\partial y} \right)^n$ Here $n < 1$ e.g. fine particles (ii) Dilatant fluids $\tau = \mu \left(\frac{\partial u}{\partial y} \right)^n$ Here $n \leq 1$ e.g. ultra-fine particles (iii) Ideal plastic or Bingham fluids $\tau = \tau_0 + \mu \left(\frac{\partial u}{\partial y} \right)^n$ e.g. water suspensions in clays	(i) Thixotropic Fluids $\tau = \mu \left(\frac{\partial u}{\partial y} \right)^n + f(t)$ e.g. crude particles (ii) Rheopectic fluids $\tau = \mu \left(\frac{\partial u}{\partial y} \right)^n + f(t)$ (iii) Rare liquid-solid suspension	Viscoelastic fluids $\tau = \mu \frac{\partial u}{\partial y} + \alpha E$ E is the modulus of elasticity. Liquid solid e.g. combinations in pipe flow and polymerized fluids with drag reduction features

1.4 Computational Fluid Mechanics: It is usually abbreviated as CFD and is defined as a branch of fluid mechanics that solves and analyses fluid flow problems, using numerically methods and algorithms. In order to perform the calculation required to simulate the fluid surface interaction, defined by boundary conditions, computers need to be employed. Advantage in employing high-speed super computers is that it provides better solutions.

1.5 Drag Coefficient: The drag coefficient is a dimensionless coefficient that is used to calculate the force of drag experienced by the object due to movement through a fully enclosed fluid.

The drag coefficient C_d is defined as:

$$C_d = \frac{2F_d}{\rho V^2 B}$$

C_d = Drag coefficient

ρ = Density of fluid

V = Flow velocity relative to object

F_d = Drag force

B = Size of the cylinder

CHAPTER 2

LITERATURE REVIEW

The following section makes a significant contribution to the understanding of the topic “Heat transfer around the square cylinder in unsteady flow regime”. The fluid dynamics study is considered for the flow around a square cylinder in a channel with the specified inlet velocity profile and governing boundary conditions. As the Reynolds number increases, the flow starts separating at rear edges of the cylinder, causing the formation of closed steady recirculation region. Such region increases with the increase in Reynolds number. The Von-Karman vortex street is observed from the representative wake behaviour as the critical Reynolds number increases. Karman Vortex Street undergoes periodic vortex shedding from the cylinder. Okajima (1982) founded periodic vortex shedding at Reynolds number approximately 70. Klekar and Patankar (1992) found out the value to be around 54. Their experimental work was based on the stability analysis of the flow separation at the leading edges between the bonds of $Re=100$ to $Re=150$ by Okajima (1982) and Franke (1991). Bruer et al (1999) used Lattice Boltzmann and Finite volume methods to model laminar flow past the square cylinder. With the rise of Reynolds number, the vortex shedding period also increases was observed by Davis et al (1984).

Franke (1990) made numerical calculations of laminar vortex shedding flow past the square and circular cylinder in laminar flow regime. He also demonstrated from hi analysis that at Reynolds number below 150, separations occur at the rear corners of the cylinder. And with the increase in Reynolds number to the threshold there is a decrease in size of the vortices. Galetti (2003) had stimulated multiple Reynolds number through proper orthogonal decomposition (POD). A brief review on the two dimensional unsteady laminar flow around a square cylinder was presented by Sharma.

Two dimensional unsteady flow around a quare cylinder was investigated by Chabra et al. Furthermore, the effects of Reynolds number and Prandlt number on the flow and isothermal patterns and local and averaged Nusselt number are discussed.

2.1 Computational fluid dynamics: CFD is a part of fluid mechanics that emphasizes on numerical scheme and algorithms to work out and study problems that are related to fluid flows. Computers are use carry out the large amount of calculations which are needed to stimulate the relations of fluid and gases with the not so easy surface that are used in engineering. Still with high speed super computers barely inexact solution can be attained. In many cases continuing study, on the other hand may give way software that give better accuracy and speed of difficult simulations situations such as transonic or turbulent flows (Acheson D.J., 1990).

2.2 Advantages of CFD:

- 1) The effect of various parameters and variables on the behaviour of the system can be studied instantaneously since the speed of computing is very high. To study the same in an experimental setup is not only difficult and tedious but also sometimes may be impossible.
- 2) Numerical modelling is flexible in nature. Problem with different level of complexity can be stimulated.
- 3) Transportation of equipment is difficult in experimental analysis, whereas in CFD software it is easy to use and modify.
- 4) CFD is also use to develop approximate analysis through governing equations of fluid mechanics in the fluid region. For such conditions CFD make use of numerical methods (called discretization).

2.2 Basic review on Heat transfer:

The transfer of heat is basically from a high-temperature object to a lower temperature object. According to the first law of thermodynamics, there is a change in the internal energy of both systems due to heat transfer.

Heat may be defined as energy in transit. When the energy is transferred to the object from a high temperature object its internal energy is increased to an extent- this condition is known as heating. Temperature is actually the measure of average translational kinetic energy which is associated with the uneven microscopic motion of atoms and molecules. Heat transfer takes place from a region of high-temperature to the region of low-temperature. Kinetic theory basically relates the relationship to molecular motion. Kinetic temperature is the temperature defined from kinetic theory. The temperature only measures the kinetic energy part of the internal energy because the temperature is not directly proportional to the internal energy. Therefore when objects possess the same temperature does not have the same internal energy.

CHAPTER 3

MATHEMATICAL FORMULATION

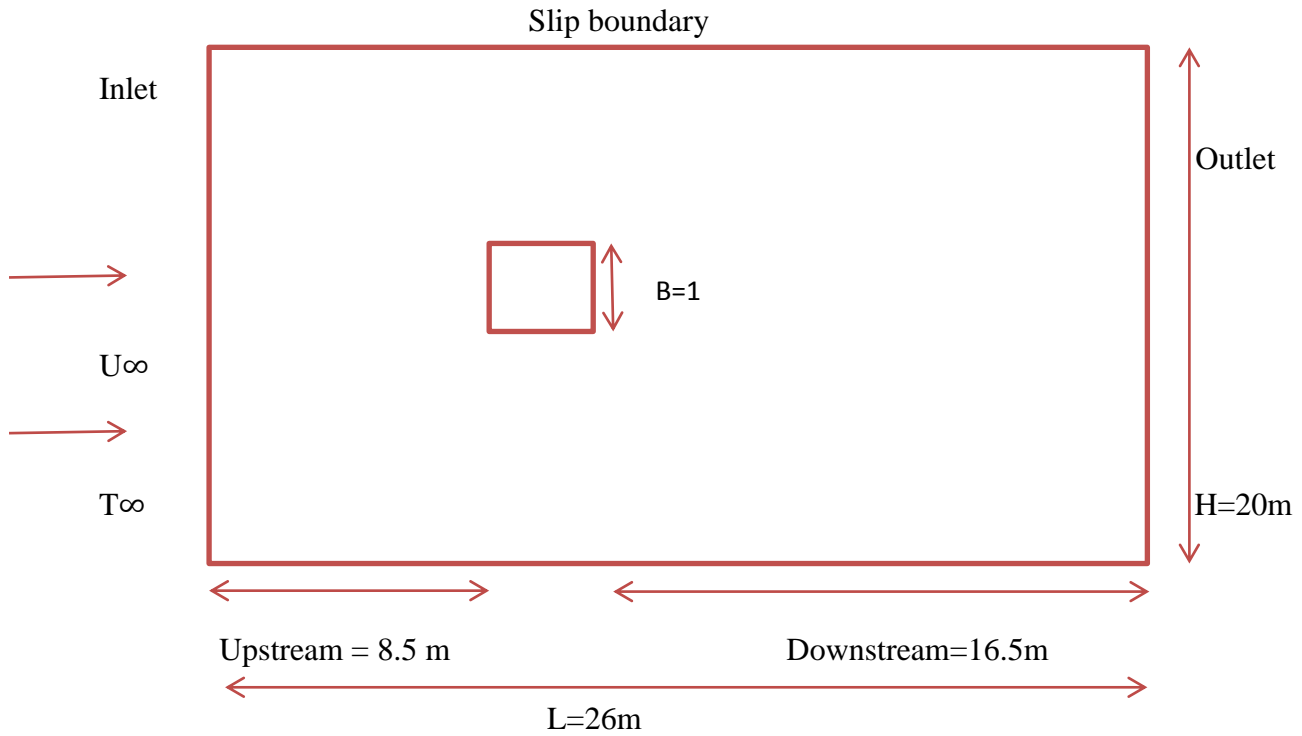


Fig. 1 Schematic flow around a Square cylinder

The flow is assumed to be two dimensional and unsteady. The flow takes place around square cylinder of size B (1m) placed in a stream of uniform velocity. The continuity equation, the momentum equations and the energy equation for the heat transfer past a square cylinder are given. Unsteady Navier stokes equation is used in solving the simulation for the fluid flow around the square cylinder.

Here the governing equations are given as:

1. Flow field:

The Continuity Equation:

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0$$

The X-Momentum Equation:

$$\frac{\partial U}{\partial \tau} + \frac{\partial(U*U)}{\partial X} + \frac{\partial(UV)}{\partial Y} = -\frac{\partial P}{\partial X} + \frac{1}{Re}(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2})$$

The Y-Momentum Equation:

$$\frac{\partial V}{\partial \tau} + \frac{\partial(V*V)}{\partial Y} + \frac{\partial(UV)}{\partial X} = -\frac{\partial P}{\partial Y} + \frac{1}{Re}(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2})$$

Here $U = \frac{u}{u_\infty}$, $V = \frac{v}{v_\infty}$, $\tau = \frac{tu_\infty}{B}$, $X = \frac{x}{B}$, $Y = \frac{y}{B}$ and $P = \frac{p}{\rho u_\infty^2}$

u_∞ is the uniform velocity of the fluid.

The boundary conditions for the flow field may be written as follows:

1) The boundary condition is set to Free slip within $y=0$ and $y=H$.

$$\frac{\partial U}{\partial Y} = 0 ; V = 0$$

2) Solid surface of the square cylinder. Here $U=0$, $V=0$.

3) The inlet boundary .Here $U=1$, $V=0$

4) At the outer boundary, the outflow is specified as the boundary condition.

2) Temperature field:

T_∞ = uniform temperature of the fluid.

Energy equation is given by:

$$\frac{\partial \theta}{\partial \tau} + \frac{\partial(U\theta)}{\partial X} + \frac{\partial(V\theta)}{\partial Y} = \frac{1}{RePr} \left(\frac{\partial^2 \theta}{\partial X^2} + \frac{\partial^2 \theta}{\partial Y^2} \right)$$

Here $\theta = (T - T_\infty) / (T_w - T_\infty)$

The boundary conditions for the temperature field may be written as follows:

1) At top and bottom walls: $\frac{\partial \theta}{\partial Y} = 0$

2) At solid surface of the cylinder: $\theta = 1$ for constant cylinder temperature and $\frac{\partial \theta}{\partial n} = -1$ for uniform heat flux.

3) At inlet boundary, $\theta = 0$.

4) At the outlet boundary, the outflow is specified as the boundary condition.

Drag coefficient:

The drag coefficient C_D is defined by

$$C_D = \frac{2F_D}{\rho U_\infty^2 B}$$

F_D = Drag Force.

ρ = Density of the fluid.

U_∞ = Speed of object relative to fluid.

B = Size of the cylinder.

Lift coefficient:

The lift coefficient C_L is defined by

$$C_L = \frac{2F_L}{\rho U_\infty^2 B}$$

F_L = Lift force.

ρ = Density of the fluid.

U_∞ = Speed of object relative to fluid.

B = Size of the cylinder.

Prandtl Number:

The prandtl number Pr is defined by

$$Pr = \frac{C_p \mu}{k}$$

C_p = Specific heat.

k = Thermal conductivity.

μ = Dynamic Viscosity.

Reynolds Number:

The Reynolds number ***Re*** is defined by:

$$\mathbf{Re} = \frac{\rho U_{\infty} B}{\mu}$$

ρ = The density of the flowing fluid.

μ = The viscosity of the flowing fluid.

B = Size of the cylinder.

U_{∞} = Characteristic velocity of the problem.

Nusselt Number:

The Nusselt number ***Nu*** is defined by:

$$\mathbf{Nu} = \frac{hB}{K}$$

h = Heat transfer coefficient.

B = Characteristic length of the problem.

k = Thermal Conductivity.

CHAPTER 4

RESULTS AND DISCUSSION

It comprises of two parts on which simulation is carried out

1. Lid-driven flow in a square cavity.
2. Heat transfer around a square cylinder in unsteady flow regime.

4.1 Problem Statement 1: To study the lid-driven square cavity using ANSYS-FLUENT (at different Reynolds number). There are 4 walls out of which three walls are stationary while one wall is a moving lid. Material here is taken as water-liquid. Flow calculations were done at different Reynolds number using the same. Variation of the x-component of velocity at horizontal centre line and y-component at vertical centre line for Newtonian fluid ($n=1$) were shown and compared the results with Ghia et al (1985). Similarly the plots for Non-Newtonian fluid at ($n=0.5$, $n=0.75$) and compared with results by Neotyfou.

1) Lid Driven Cavity

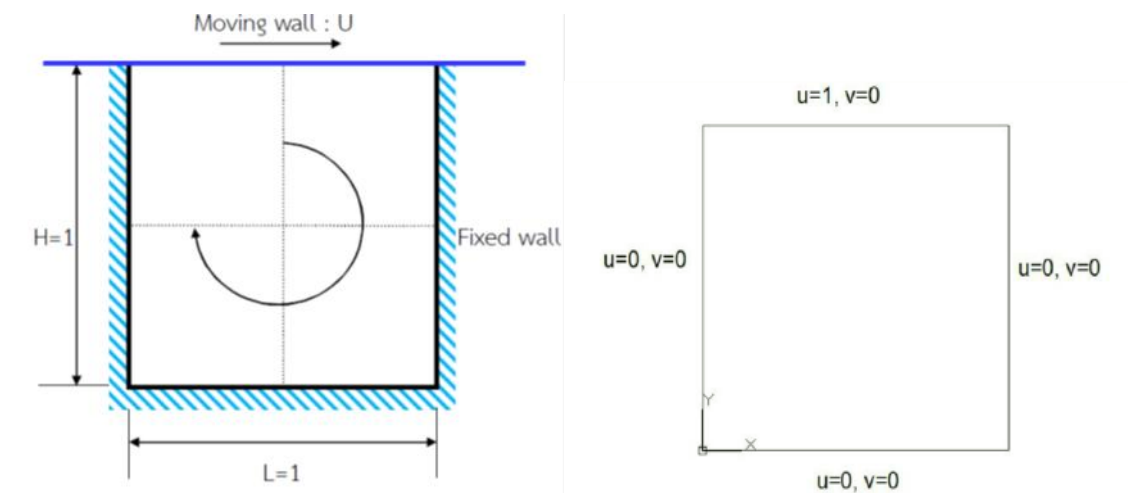


Fig. 2 Lid driven cavity

SPECIFICATION:

- 1) Fluid flow inside a $1 \times 1 \text{ m}^2$ square cavity as shown in figure.

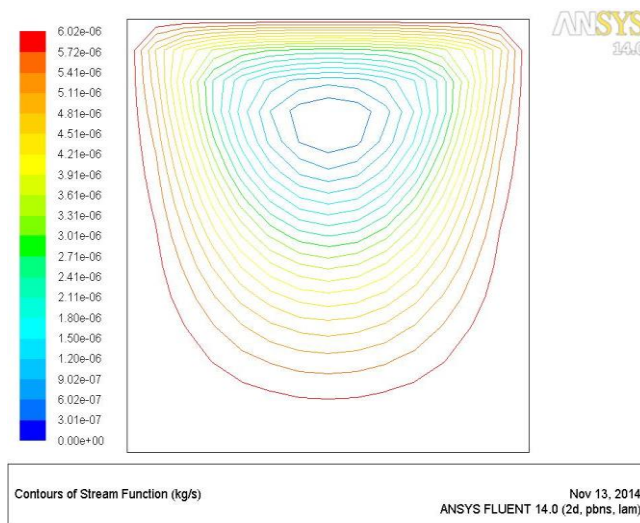
2) Upper wall moving with constant velocity of $U=1\text{m/s}$.

3) Reynolds number based on the cavity height can be calculated.

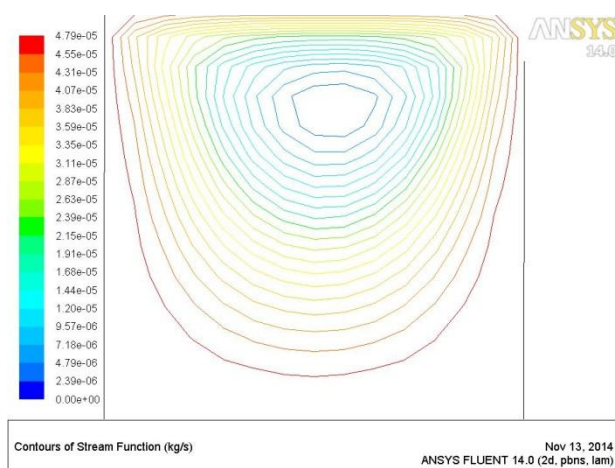
$$\text{Re} = \frac{\rho U D}{\mu}$$

Here $\rho = 998.2\text{kg/m}^3$, $\mu = 0.001003\text{ kg/ms}$, Thermal conductivity $K = 0.6\text{ W/mk}$, Specific heat capacity = 4182 J/kgk

2) Stream function plots for various Reynolds number for Newtonian fluid:

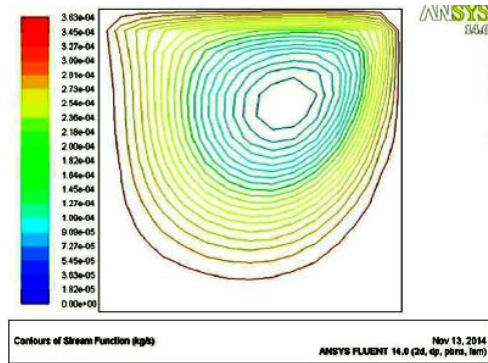


(a) $\text{Re}=100$

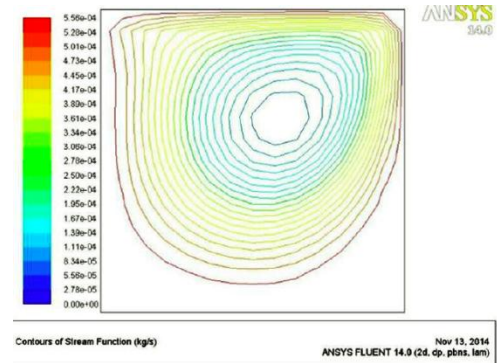


(b) $\text{Re}=400$

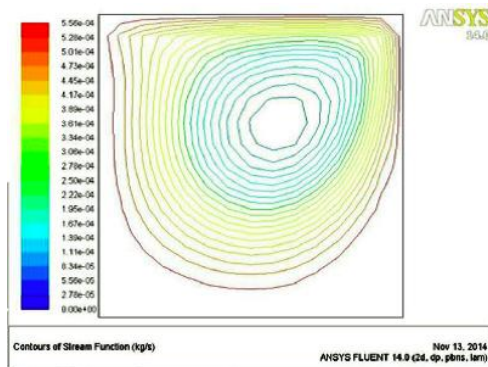
Fig. 3 Stream Function plots at (a) $\text{Re}=100$ and (b) $\text{Re}=400$.



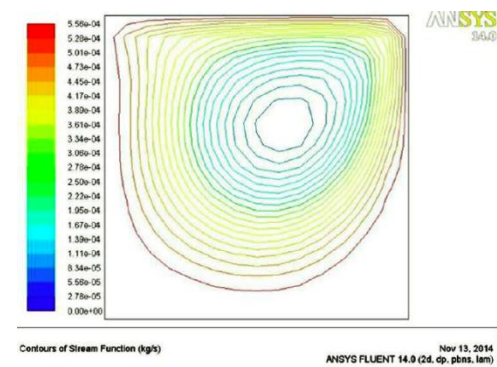
(c) $Re=3200$



(d) $Re=5000$



(e) $Re=7500$



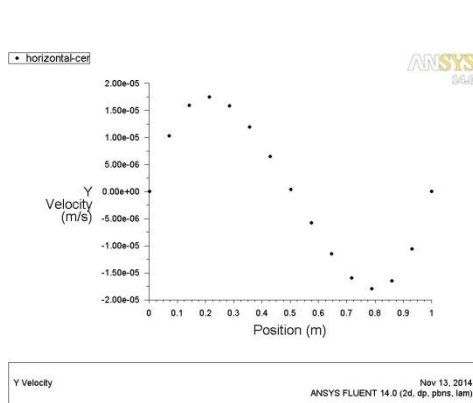
(f) $Re=10000$

Fig. 3 Stream Function plots at (c) $Re=3200$, (d) $Re=5000$, (e) $Re=7500$ and (f) $Re=10000$.

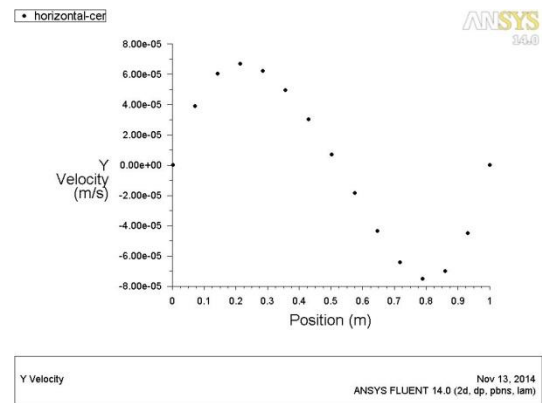
We had taken Newtonian fluid at Reynolds number from 100 to 10,000. From the Stream line plots shown in the above study we see that at Reynolds number around 100, the primary vortex moves towards the right hand wall and the downstream secondary eddy starts to enlarge in size.

At $Re=400$, the primary vortex start moving towards centre even in high Reynolds number. As we increase the Reynolds number the vorticity gradients develop on the lid and cavity walls for higher Reynolds number.

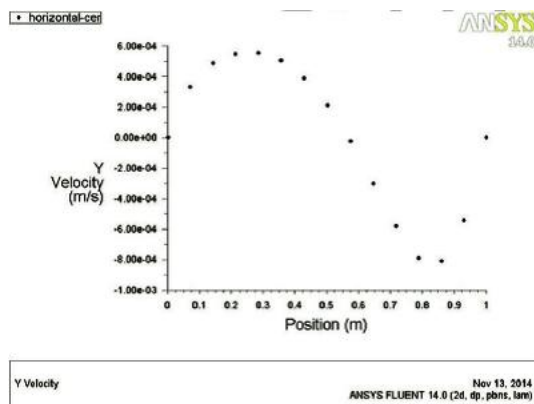
3) Horizontal centre line velocities at various Reynolds number for Newtonian fluid:



(a) $Re=100$

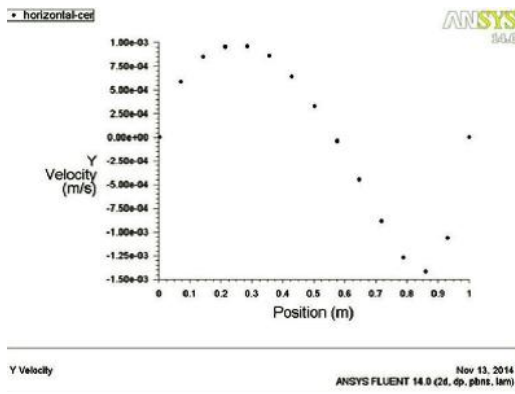


(b) $Re=400$

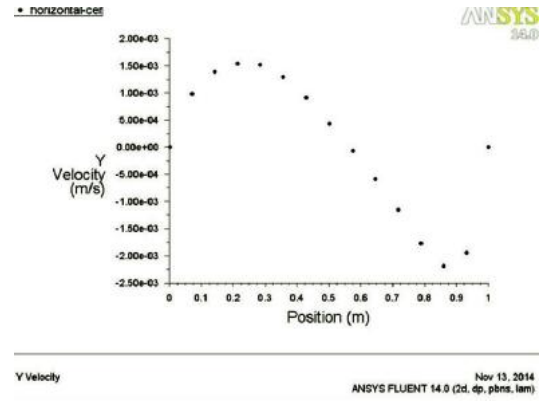


(c) $Re=3200$

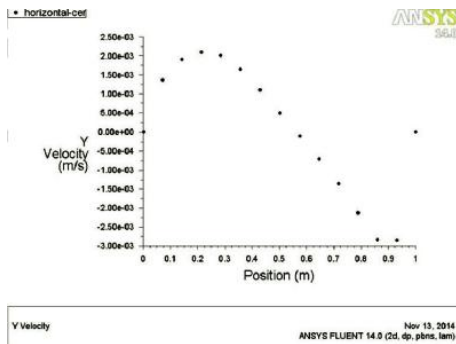
Fig. 4 Horizontal center line velocity at (a) $Re=100$, (b) $Re=400$ and (c) $Re=3200$.



(d) $Re=5000$



(e) $Re=7500$

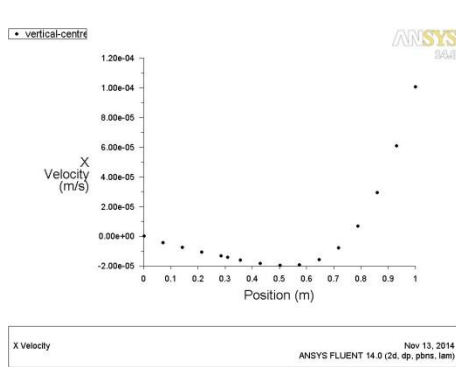


(f) $Re = 10,000$

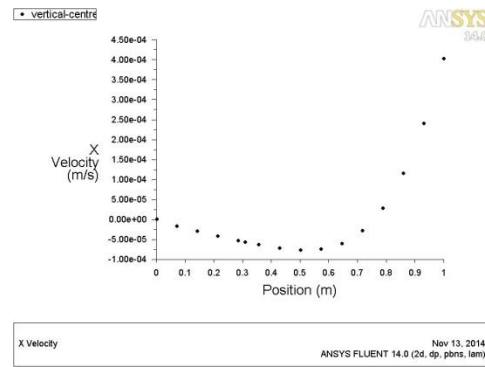
Fig. 4 Horizontal center line velocity at (d) $Re=5000$, (e) $Re=7500$ and (f) $Re=10,000$.

Horizontal centre line velocity has been calculated for various Reynolds number ranging from 100 to 10,000 in order to observe the flow behaviour.

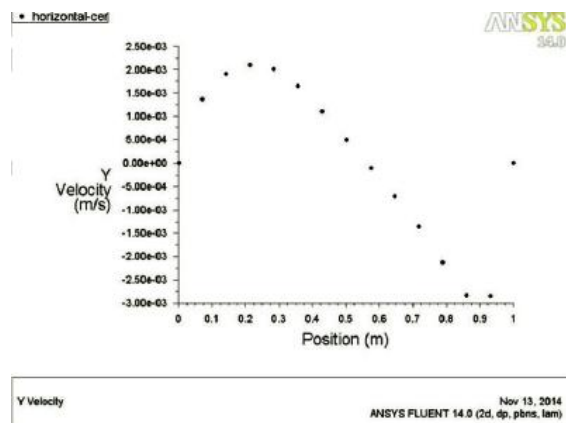
4) Vertical centre line velocities at various Reynolds number for Newtonian fluid:



(a) Re=100

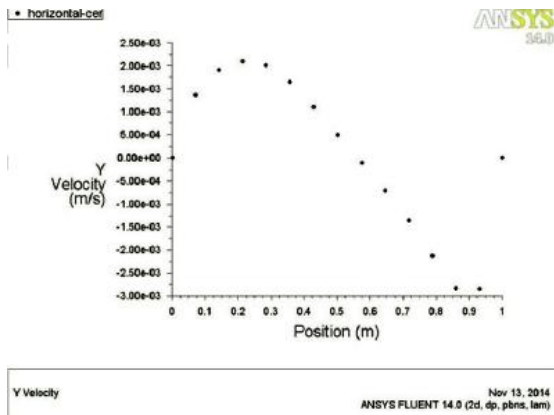


(b) Re=400

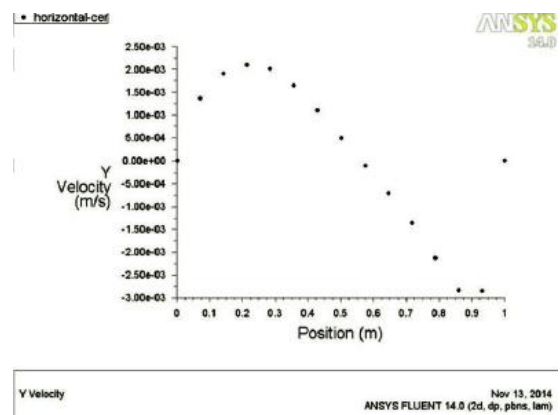


(c) Re=3200

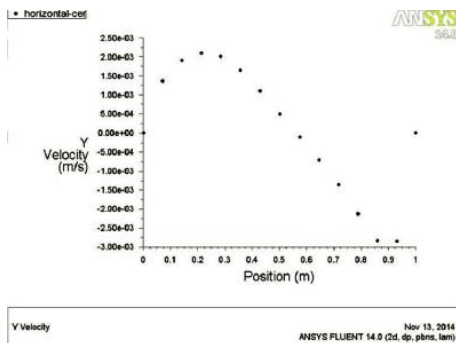
Fig. 5 Vertical center line velocity at (a) Re=100, (b) Re=400 and (c) Re=3200.



(d) $Re=5000$



(e) $Re=7500$



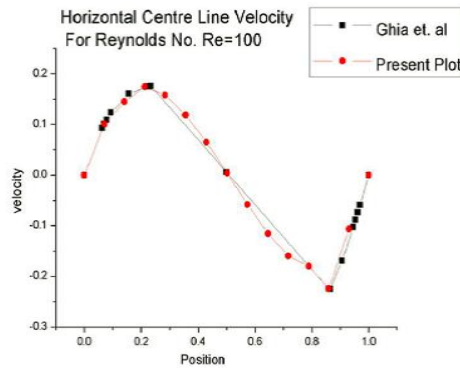
(f) $Re=10000$

Fig. 5 Vertical center line velocity at (d) $Re=5000$, (e) $Re=7500$ and (f) $Re=10,000$.

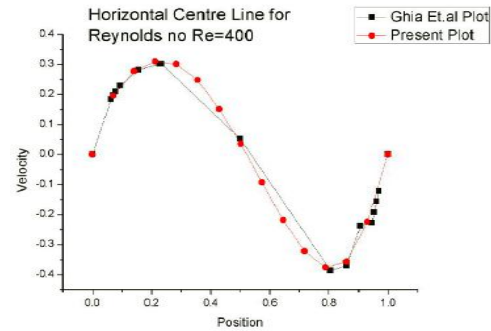
Vertical centre line velocity has been calculated for various Reynolds number ranging from 100 to 10,000 in order to observe the flow behaviour.

5) Comparison of results of Horizontal centre line and Vertical centre line for $Re=100$ & 400 by Ghia et al for Newtonian fluid:

1. Horizontal centre line



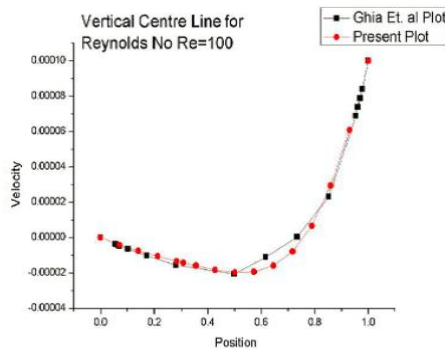
(a) $Re=100$



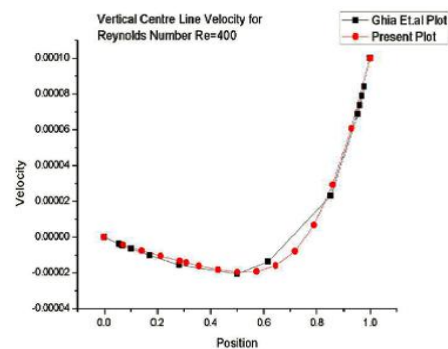
(b) $Re=400$

Fig. 6 Horizontal center line velocity at (a) $Re=100$ and (b) $Re=400$.

2) Vertical centre line



(a) $Re=100$



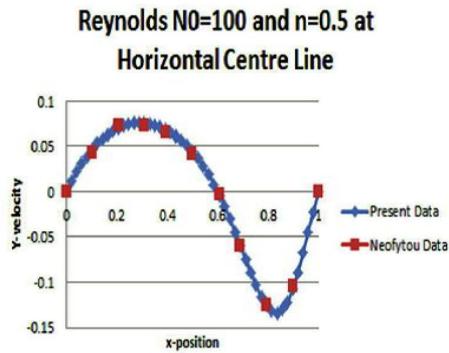
(b) $Re=400$

Fig. 7 Vertical center line velocity at (a) $Re=100$ and (b) $Re=400$.

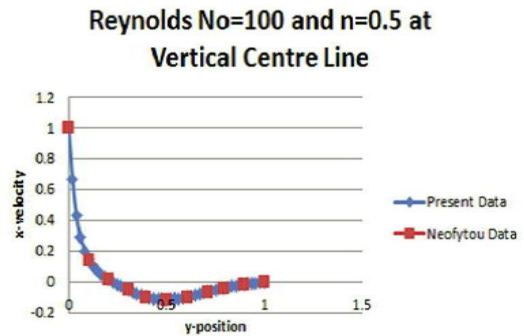
For an assessment the accuracy of the present results, the velocity components through the vertical and horizontal centre lines are compared with the corresponding results of the Ghia et al (1982). The comparison shows good agreement, particularly at Reynolds number up to 5000. But at Reynolds number 10,000 the present values gives slightly higher external values of the velocity components.

6) Comparison of results of Horizontal centre line and Vertical centre line for $Re=100$ by Neofytou for Non-Newtonian fluid:

1) $N=0.5$



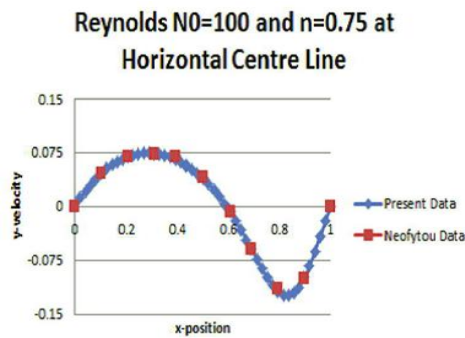
(a) $Re=100$



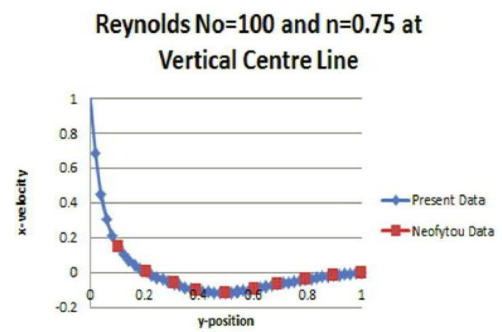
(b) $Re=100$

Fig. 8 Horizontal center line velocity at (a) $Re=100$ and Vertical center line velocity at (b) $Re=100$ for $n=0.5$.

2) $N=0.75$



(a) $Re=100$



(b) $Re=100$

Fig. 9 Horizontal center line velocity at (a) $Re=100$ and Vertical center line velocity at (b) $Re=100$ for $n=0.75$.

Similarly, for Non-Newtonian flow we had shown the vector plots for the same Reynolds number. The plots show the onset formation of the wakes at the bottom corners of the cavity. The velocity components were compared with the results of Neofytou at $Re=100$, and it showed good alignment.

4.2) Problem Statement 2: The main objective of this simulation is to study the heat transfer around a square cylinder at various conditions taking Reynolds number (90,100,120) and Prandlt number (0.7, 5, and 10). Problem domain is a rectangular computational around a square cylinder of 1m dimension in it. The vortices and temperature evolutions show the flow and temperature field change and vortex shedding plays a determinant role in heat transfer. Furthermore, the effects of Reynolds number and Prandlt number on the flow and isothermal pattern and local and average Nusselt number are to be calculated.

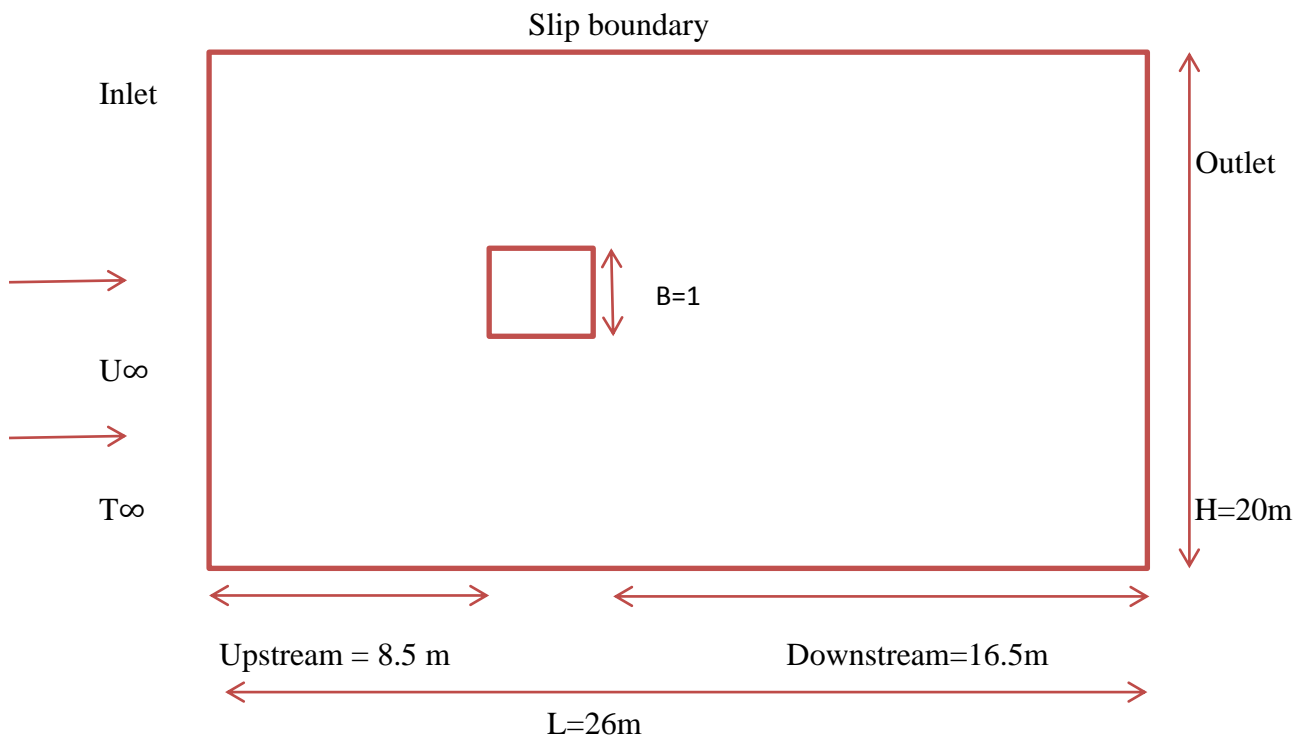


Fig. 10 Flow over a square cylinder

PROCEDURE:

1. Create Geometry:

Here we are using 2D geometry analysis. The length units are set to meters. The computation has been carried out using grid size of 211x137. Square cylinder is kept inside the rectangular computational domain. Here we are considering upstream =8.5m and downstream=16.5m from the walls as shown in the figure.

2. Meshing the geometry in the ANSYS meshing application:

The unstructured cells of non-uniform spacing were generated.

The number of nodes distributed on each surface of the cylinder= 100.

The number of nodes for top and bottom walls =400.

The number of nodes for inlet and outlet=200.

3. Setting up the CFD simulation in ANSYS FLUENT:

The mesh generated is then sent to the fluent solver for further simulation

1. Set up some general setting in the CFD analysis:

Here in the solver option we select type as pressure based, plan as implicit, and space as 2D. The time is selected as unsteady as given in the problem statement.

2. Set up models for CFD analysis:

In the models option select viscous → laminar → Ok

Set Energy → On

3. Set up materials for CFD analysis:

Here the density was taken to be 1Kg/m³ and the viscosity was changed according to the Reynolds number.

4. Set up boundary conditions for CFD analysis:

1) At top and bottom wall: Respective symmetry is defined for both the top and bottom walls. The walls are set to no slip condition. For unsteady flow, “wall” was selected and no shear condition was selected.

2) At inlet: The inlet boundary was set as “velocity inlet”.

Outlet: The outlet boundary was set as “pressure outflow” for unsteady flow over a square cylinder.

The reference values were calculated from the inlet.

5. Set up the parameters for CFD analysis:

The discretization methods used for momentum are the pressure interpolation scheme and QUICK scheme. A segregated solver with Simple algorithms was used to obtain results through undergoing series of iterations. Second order transient was selected for unsteady flow. Both residual monitor and convergence criteria were taken to be around 1e-6. Drag and lift monitors were set to obtain plots. All the values were saved through the “write” option. Patching was done with uniform velocity in x=direction of 1m/s in upper part of domain and 0m/s in lower part of domain.

$$u = \frac{y+|y|}{2y}$$

u is the x velocity.

6. Run Calculation: For unsteady flow, Number of Time step=0.3s, Maximum time step=100 and Number of iterations= 1000.

Results obtained:

Table no.1 Values of Cd and Nu at Pr=0.7

Re	Cd	Nu
90	1.53	15.58
100	1.50	16.27
120	1.48	17.68

Table no.2 Values of Cd and Nu at Pr=5

Re	Cd	Nu
90	1.53	31.91
100	1.51	33.26
120	1.48	36.1

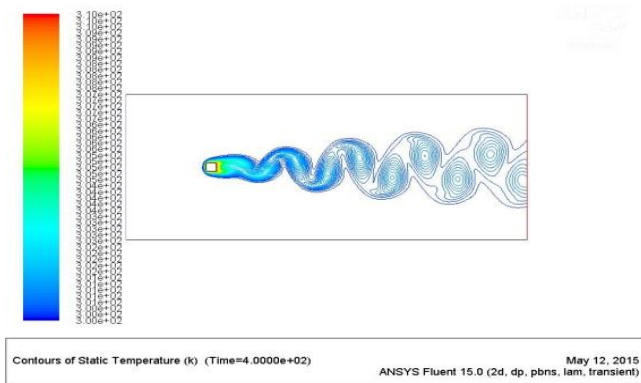
Table no.3 Values of Cd and Nu at Pr=10

Re	Cd	Nu
90	1.53	40.9
100	1.57	42.61
120	1.48	44.27

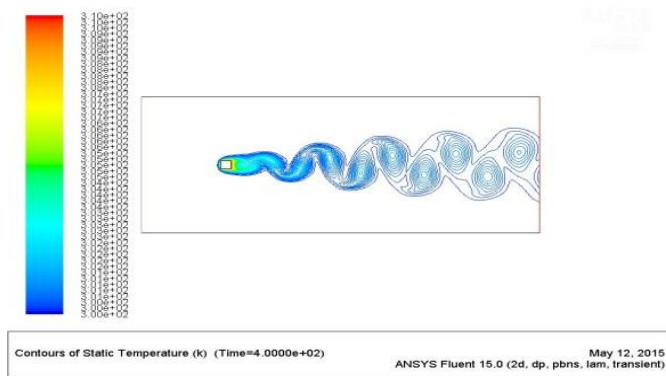
Table no.4 Values of Cd and Nu at Pr=20

Re	Cd	Nu
90	1.53	52.29
100	1.51	54.42
120	1.48	58.65

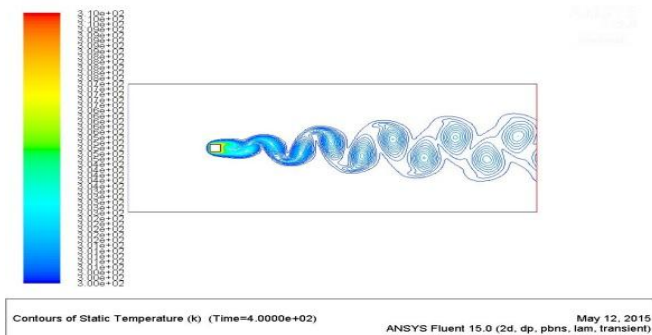
Isothermal pattern at $Pr=0.7$ for various Reynolds Number:



(a) $Re=90$



(b) $Re=100$

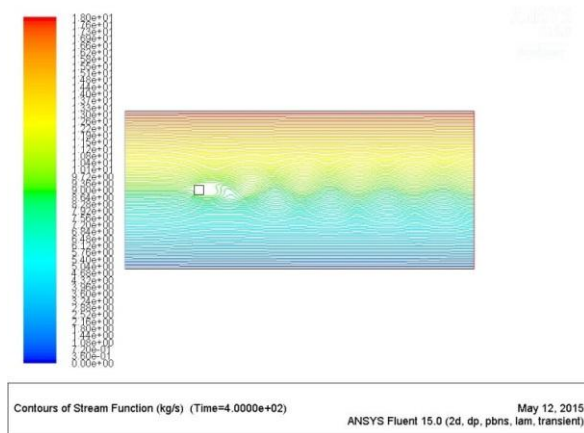


(a) $Re=120$

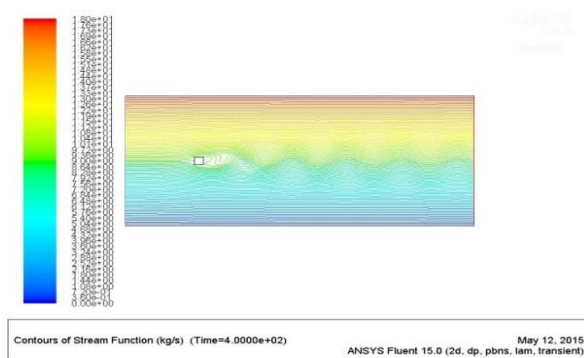
Fig. 11 Isotherm pattern for $Pr=0.7$ at (a) $Re=90$, (b) $Re=100$ and (c) $Re=120$

The representative instantaneous isotherms near the square cylinder are presented for the Reynolds Number 90,100 and 120. The figure reveals that a temperature street is formed downstream the square cylinder.

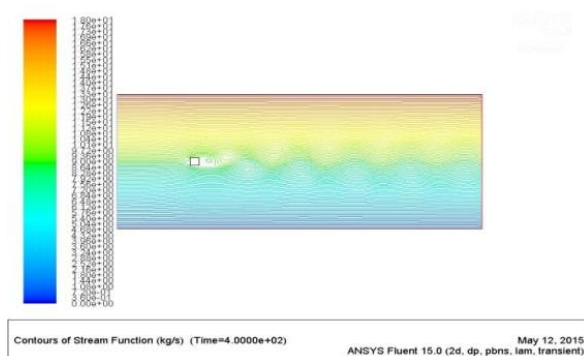
Stream function at $Pr=0.7$ for various Reynolds number:



(a) $Re=90$



(b) $Re=100$

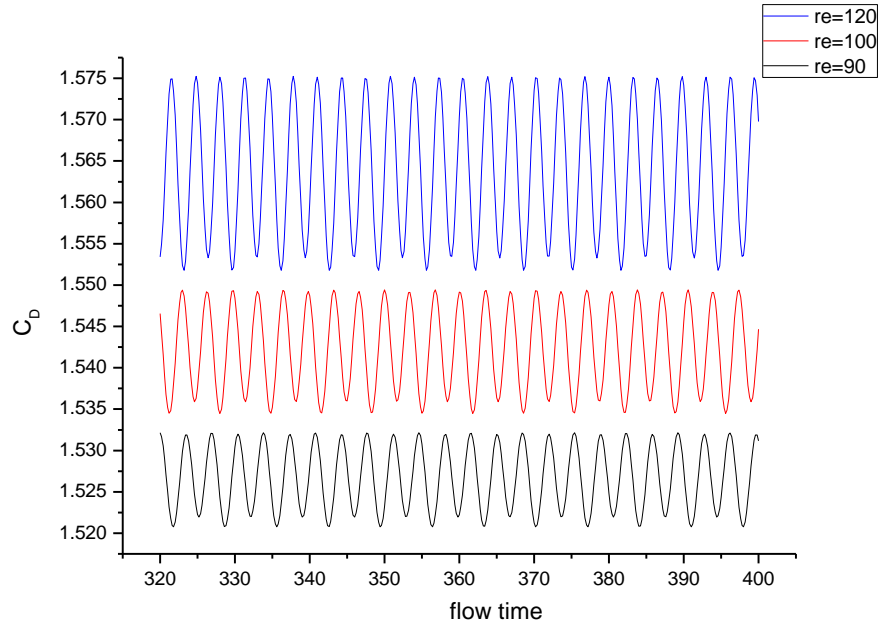


(c) $Re=120$

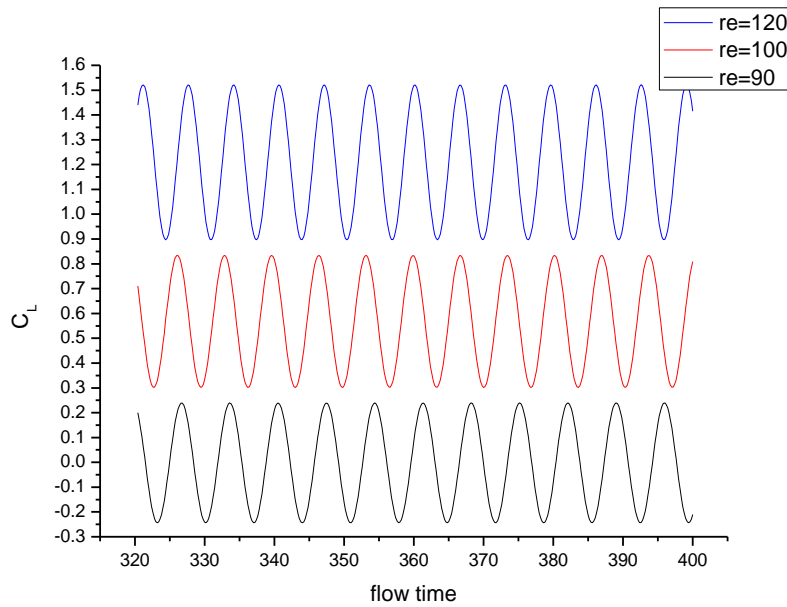
Fig.12 Stream function plots for $Pr=0.7$ at (a) $Re=90$, (b) $Re=100$ and (c) $Re=120$

The stream function plots for Reynolds number 90,100 and 120 are shown. The recirculating eddies are formed immediately downstream of the cylinder as indicated by the streamlines.

Drag coefficient and Lift coefficient variation on square cylinder for different Reynolds number at $Pr=0.7$:



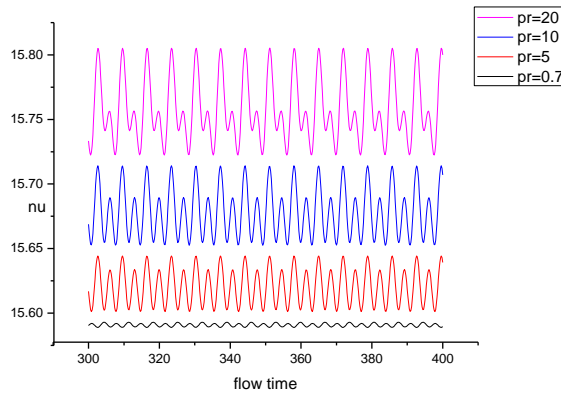
(a)



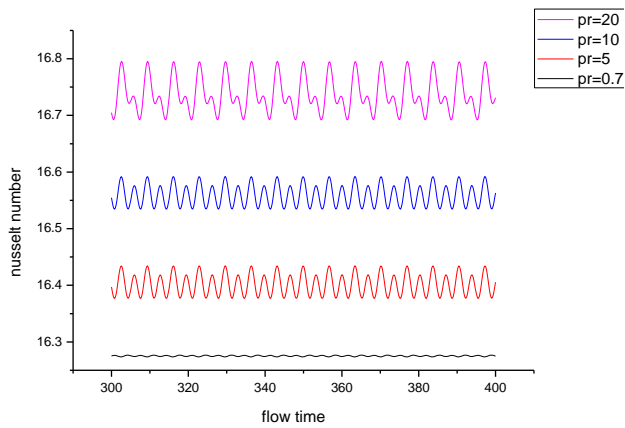
(b)

Fig. 13 Variation of (a) drag coefficient and (b) lift coefficient for $Re=90, 100$ and 120 at $Pr=0.7$.

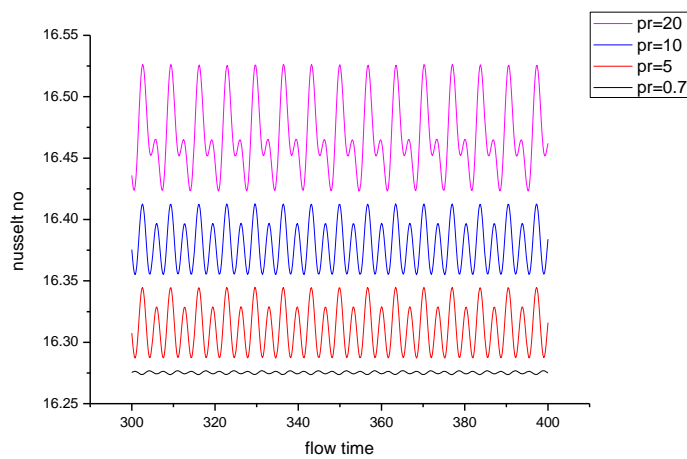
Nusselt number variation on square cylinder surface at different Prandlt number:



(c) Re=90



(b) Re=100



(c) Re=120

Fig. 14 Variation of Local Nusselt number at different Prandlt number for (a) Re=90, (b) Re=100 and (c) Re=120.

Average Nusselt number variation on square cylinder for different Prandlt number:

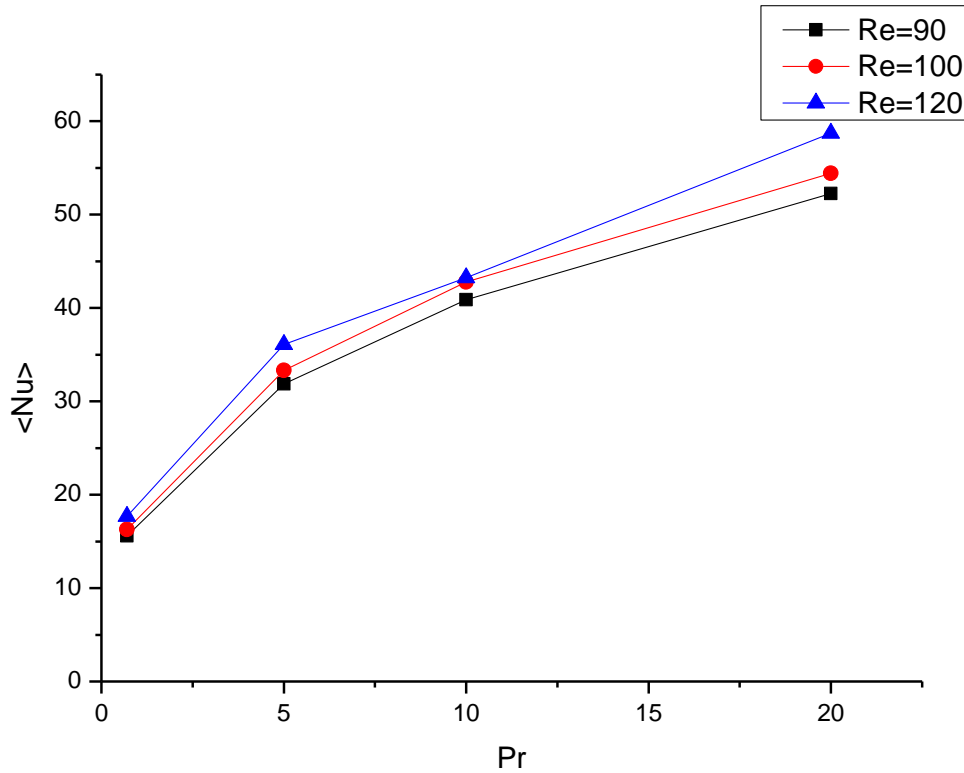


Fig. 15 Variation of average Nusselt number with Re and Pr

For both steady and unsteady flow the Nusselt number increases with Prandlt number and the highest value is observed at the front stagnation. Nusselt number is the ratio of the convective to conductive parts of total heat transfer and, the higher Prandlt number flow indicates a strong convective effect of heat transfer; therefore a high Prandlt number should be associated with high Nusselt number. The following figures from Fig. 39 to Fig. 41 represents that for unsteady flow regime the local Nusselt number is almost monotonous at low Prandlt number. Fig. 42 represents the effect of Reynolds and Prandlt number on the surface averaged Nusselt number. The average Nusselt number shows an increase with Reynolds and Prandlt number.

CHAPTER 5

CONCLUSIONS

The flow through a lid driven square cavity is simulated for 2D case for different Reynolds number using ANSYS. The cavity flow problem serves a good example for different numerical methods and boundary conditions associated with accuracy, convergence rate etc.

The Heat transfer through a square cylinder in unsteady flow regime is simulated for Newtonian fluid. The flow pattern is presented via instantaneous streamline, vorticity magnitude, and velocity magnitude and pressure profile. The present results show that the local Nusselt number as well as the averaged Nusselt number increases with the Reynolds number.

REFERENCES

1. U. GHIA, K. N. GHIA AND C. T. SHIN, “High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method” JOURNAL OF COMPUTATIONAL PHYSICS 48,387-411 (1982).
2. Panagiotis Neofytou, “A 3rd order upwind finite volume method for generalised Newtonian fluid flows” Advances in Engineering Software 36 (2005) 664–680.
3. P. Koteswara Rao, C. Sasmal, A.K. Sahu, R.P. Chhabra, V. Eswaran, “Effect of power-law fluid behaviour on momentum and heat transfer characteristics of an inclined square cylinder in steady flow regime “International Journal of Heat and Mass Transfer 54 (2011) 2854–2867.
4. C. F. Lange, F. Durst and M. Breuer, Momentum and heat transfer from cylinder in laminar cross-flow at $10 \leq Re \leq 200$, Int. J. Heat Mass Transfer, 41 (1998), 3409-3430.
3. R. Byron Bird, Warren E.Stewart, Edwin N. Lightfoot; Transport Phenomena Second Edition; John Wiley & Sons, INC.; 2007;978-0-470-11539-8.
4. A.K.Saha, G.Biswas and K.Muralidhar “Three dimensional study of flow past a square cylinder at low Reynolds”, International Journal of Heat and Fluid Flow 24(2003) 54-66.
- 5 Kelkar, K.M.; and Patankar, S.V., Numerical Prediction of vortex shedding behind a square cylinder, International journal , Numerical methods in Fluids vol(14) 1992,327.